

CNC Router

Introduction

March 2023

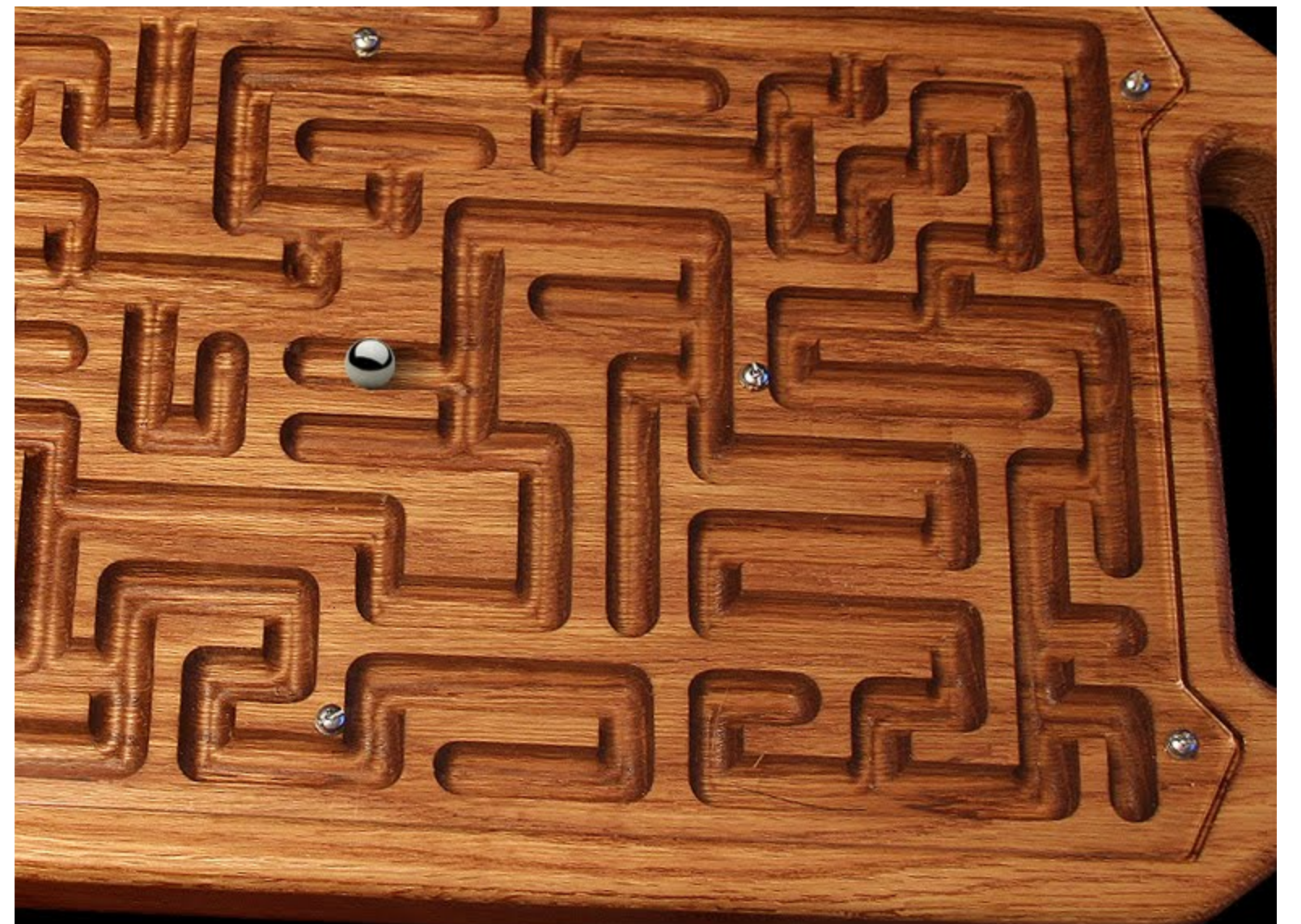
Creative
DUNBLANE

What can you make on a CNC?

Section 1





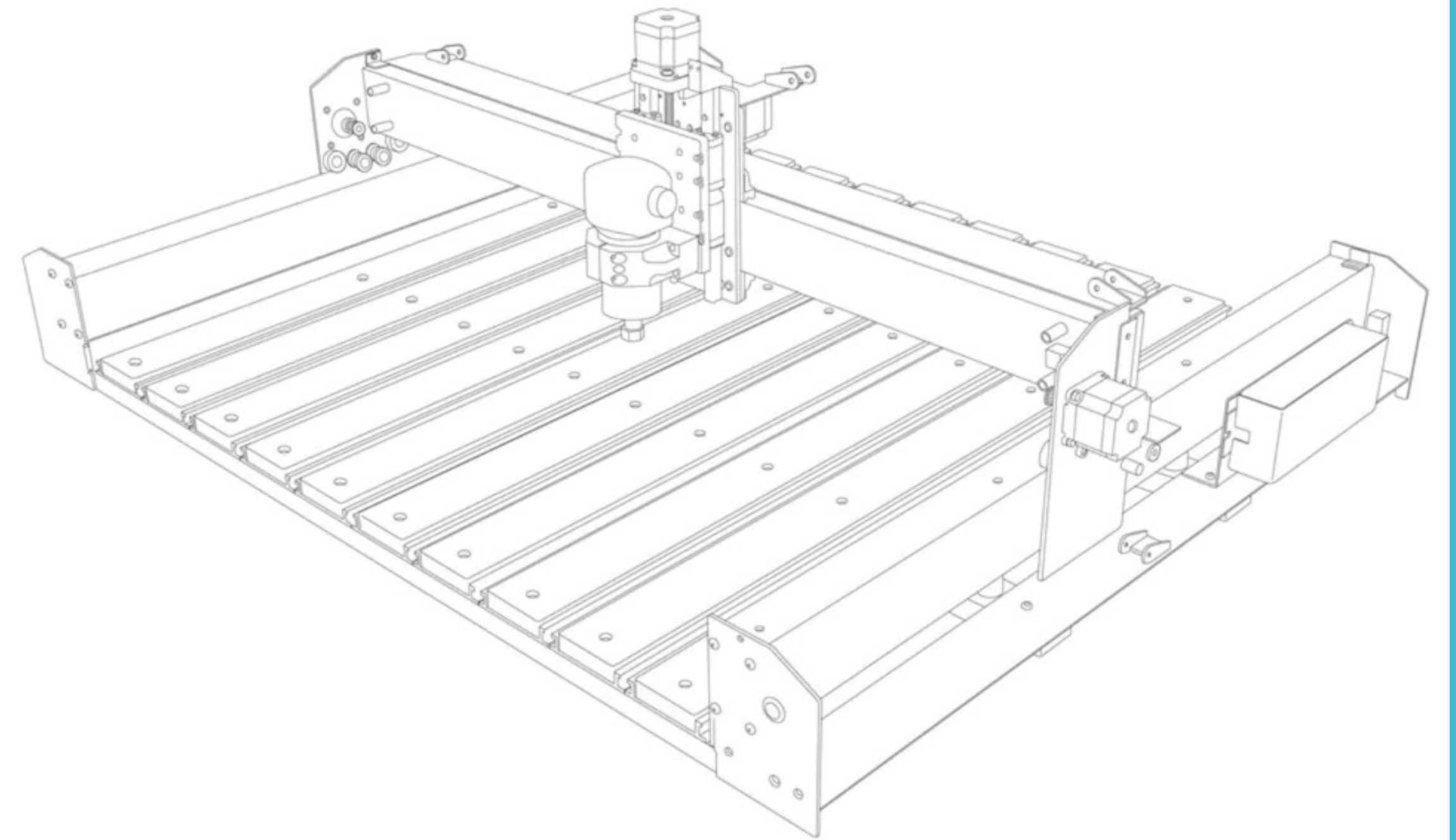




Creative
DUNBLANE

CNC Router Basics

Section 2



CNC Router Basics

What is a CNC Router?

A **computer numerical control (CNC) router** is a computer-controlled cutting machine which typically mounts a hand-held router as a spindle which is used for cutting various materials.

CNC routers can perform the tasks of many carpentry shop machines such as the panel saw, the spindle moulder, and the boring machine. They can also cut joinery such as mortises and tenons.

A CNC router is very similar in concept to a CNC milling machine. Instead of routing by hand, tool paths are controlled via computer numerical control. The CNC router is one of many kinds of tools that have CNC variants.

A CNC router can be used to produce items such as door carvings, interior and exterior decorations, wood panels, sign boards, wooden frames, moldings, musical instruments, furniture. In addition, they see use in industry in the thermoforming of plastics by automating the trimming process. CNC routers can help ensure part repeatability and sufficiently efficient output for production, or allow one-off designs to be made.

CNC Router Basics

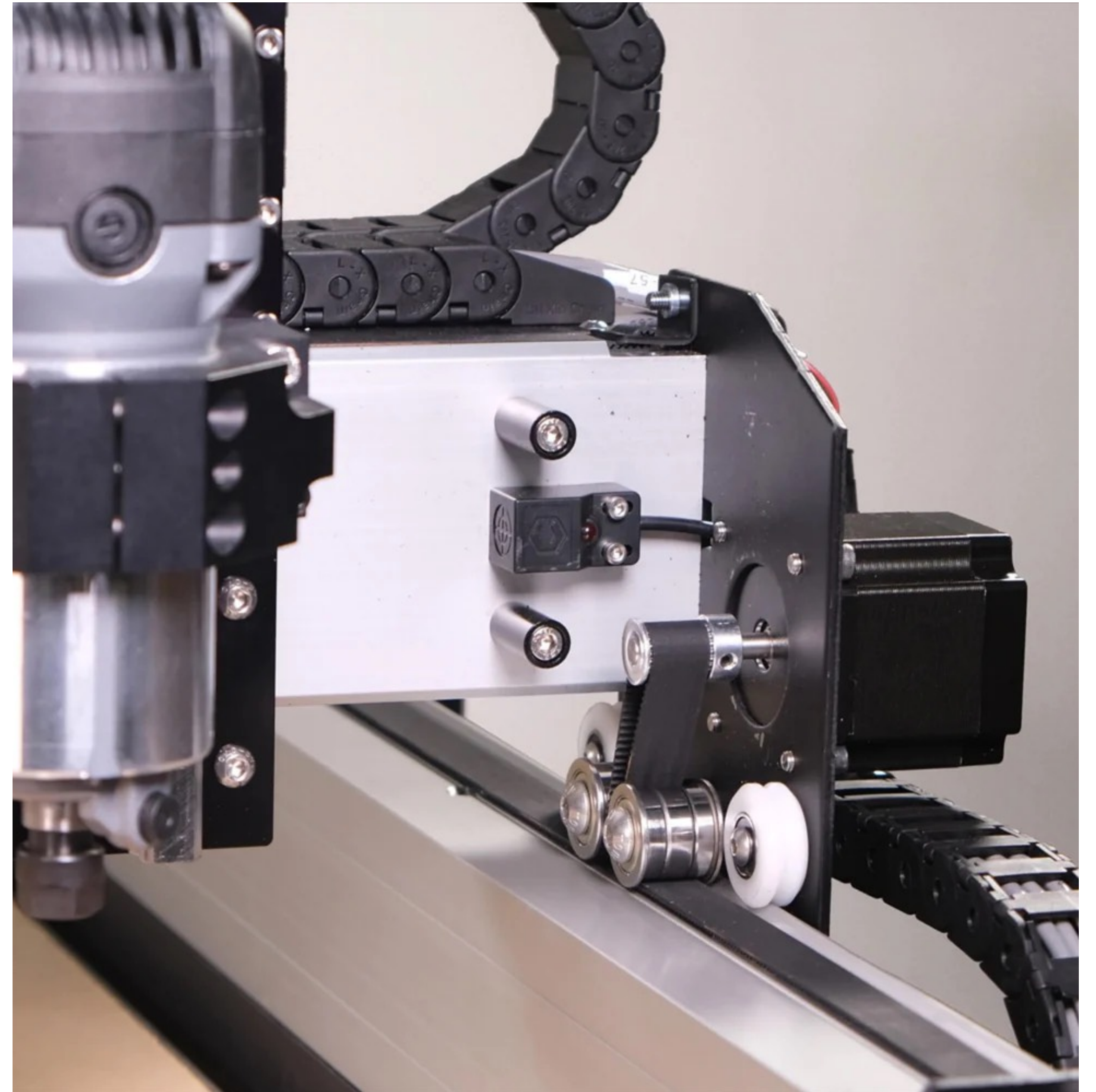
How does it work?

CNC routers are controlled by a computer. Coordinates are uploaded into the machine controller from a separate program. CNC routers are often used with two software applications—one to make designs (CAD) and another to translate those designs into a G-code or M-code program of instructions for the machine (CAM) in vertical, horizontal and perpendicular coordinates.

As with CNC milling machines, CNC routers can be controlled directly by manual programming, but CAD/CAM allows wider possibilities for contouring, speeding up the programming process and in some cases creating programs whose manual programming would be impractical.

The human operator selects the machine tool (such as a 1/4-inch (6-MM) v-bit or a 3/4-inch core box bit), speed, cut depth and tool path. For cut path, most machines give the options of tracing the vectors, cutting outside the vectors, or cutting inside the vectors.

CAM (Computer Aided Manufacturing) software makes the CAD drawing/design into a code called G-code. The illustration shows what a bare-bones CNC machine might look like without its computer controller.



CNC Router Basics

Parts of a CNC Router

1. Frame (Supports the Machine)

It's critically important that the frame of a CNC router is as stiff as it can be. Precision is impossible if the position of the workpiece or the cutter is uncertain because of the flexure of the frame. The consequences of an insufficiently stiff structure in a router vary, depending on the degree of flexibility, including: chatter, cross-bridge flexing, and displacements on any one axis.

2. Table (Spoilboard or Wasteboard)

Tables vary in their construction and locating systems for workpieces. The most adaptable tables are those made with vacuum pull-down or integrated T slots (or both), so a wide variety of workpieces can be clamped anywhere. Aluminium extrusion construction is very common and offers a good mix of adaptability, stiffness, and low weight. Alternatively, a range of strong materials can be machined to make a T-slot table with slots on two axes, making a checkerboard pattern of mount points.

3. Spindle (Heart of any CNC Machine)

The spindle is the single most critical component of a CNC router. In most machines, the spindle is integral to the motor shaft and relies on the motor bearings. This is a low-cost construction method, but unless the motor bearings are sufficiently robust, premature wear problems may result.

Spindle/motor bearings come in various qualities; they wear items that require periodic replacement. Loose bearings mean a loose cutter, and accuracy is lost. Stiff bearings add hard work and more heat to the process. Bad quality or worn bearings add a lot of noise to a fast-turning cutter.

CNC Router Basics

Parts of a CNC Router

4. Ways (Sliding Ways)

The *ways* or slides are the motion tracks that enable X-Y-Z movement. It is from these (and a stiff frame and good quality lead screws) that any router derives its precision ...or lack of it. The most common *ways* are precision-ground stainless steel rods on which either plain bushes or recirculating ball collars run. An axis requires at least two such rods – and the further apart those rods are, the more precise the axis motions will be. Plain bushes are lower cost, recirculating ball bushings are freer moving but more expensive. Free motion means more precise positioning.

More expensive machines tend to employ V-form tracks with grooved, sealed ball races locked onto both sides of the track. This track provides equal stiffness but requires less height to maintain the motion. Recirculating ball slides are harder to protect from dirt than sealed ball races. They have wiper seals that clean the ground shaft way, but some material will often get past this.

5. Axis Drive (Converts Rotary Motion)

The axis drives convert the output of the stepper motors that drive the machine from rotary to linear motion. Lower-cost machines often use toothed or timing belts for this. While that may sound less than optimal, toothed belts have been used in robotic positioning systems in this way for decades and can be durable, when kept clean.

More expensive machines tend to have recirculating ball screws. These offer the advantages of freer motion and lower hysteresis. But they pose the same contamination/wear issues that similar systems in the sliding ways suffer, so cleanliness is essential for long machine life.

6. CNC Controller (Executes G-Code)

The control computer module is the brain of any CNC machine. It takes G-code and turns it into motion by driving/controlling the various axis stepper motors and the spindle motor. These devices are sophisticated I/O control devices that control a range of outputs – from simple 3-axis and cutter speed up to 5-axis, with coolant pumps, motor cooling systems, automated cutter/router changers, and more.

CNC controllers are generally single-board computers. They are composed of: processors, memory, integrated self-protection mechanisms, and power supply. CNC controllers are designed to turn the input G-code the user provides into precise and repeatable movements that cut a complex net shape from a material blank.



1. Frame

2. Table

3. Spindle

4. Ways

5. Axis Drive

6. Controller

CNC Router Basics

CNC Router Accessories

1. CNC Router Bits

“Bits” is a catch-all term for a wide range of drills and cutters. This accessory includes: the face or shell mill, square and bull nose end mills, and ball end mill. Bullnose and ball end mills are good for cutting the surfaces of curves, as they don’t create grooves and can blend surfaces to a smooth roundness.

2. CNC Collet

A collet is a simple clamping system that uses a split tube (with a tapered nose). It forms a close fit with the straight tool shank and has a lock nut that clamps the taper to squeeze the split tube onto the tool. The collet will sit within a tool holder, often called a collet chuck, which is generally fitted to the router with a taper locator and spring locator to lock it in place. In many simpler setups, the collet chuck is not removed from the spindle but is fixed in place, so new tools and collets to suit them are handled in place.

3. Dust Extractor

To remove the dust and wood shavings from the machine it is used in conjunction with a dust extractor. The extractor is under the machine table and is fed by a tube from the top of the cabinet.

Health and Safety

Section 3



Health and Safety

Risks and Mitigation

Mechanical Hazards

Mechanical hazards involve moving machine parts and ways that a worker's body could come in contact with those parts. We try and mitigate this risk by having the CNC in an enclosed cabinet when running. Make sure the power is off before performing any maintenance and turn off power to the spindle/router when changing bits.

Health Hazards

Along with potential physical injuries caused by moving parts, workers could have exposure to hazards through inhalation, ingestion, skin contact, or absorption through the skin. We do not use chemicals with the CNC router but it does generate a lot of dust. Make sure the dust extractor is running during operation.

Other Hazards

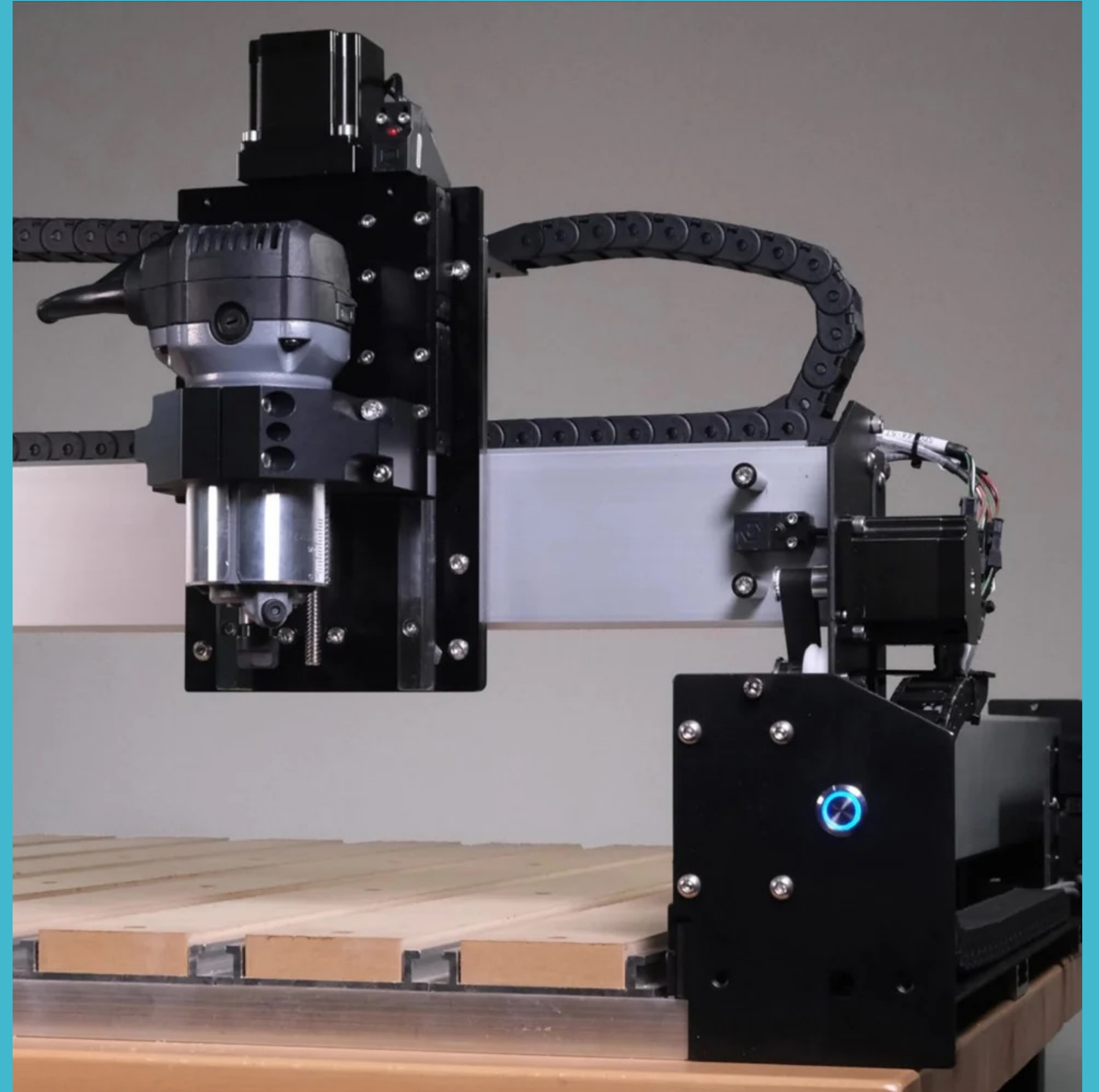
Several other dangers exist in machine shops. It is important to identify and mitigate the risks around these hazards. Slips and falls around machinery, injuries from unstable equipment, and faulty or ungrounded electrical components all pose significant dangers. Fire risks are also present in machine shops and depending on the type of equipment or machining operations, CNC machines could also be at a higher risk of fire. Many of these other hazards are addressable with good housekeeping measures and having processes for regular equipment inspections to keep the shop floor and employees safe.

For slips and falls, ensuring proper signage during maintenance, requiring suitable footwear, and keeping the floor free of debris is essential. Securing and stabilising equipment will help alleviate the risk of injuries, and making sure electrical components and sockets are functioning properly will help protect from electrical shocks or electrocution.

Creative
DUNBLANE

Our CNC Router

Section 4



Our CNC Router

Specifications

CNC Router Shapeoko 4 XXL

Router (Spindle) Makita RT0700C

End Mills #101,#102,#201,#202,#251,#301,#302,#112

Link to machine <https://shop.carbide3d.com/products/shapeoko4?variant=33051555987517>

Link to router <https://www.youtube.com/watch?v=Z-UtqryLrZg>

Power output

710 watt

Spindle speed

10,000-30,000 rpm

Cutting Area

33"(X) x 33"(Y) x 4"(Z)

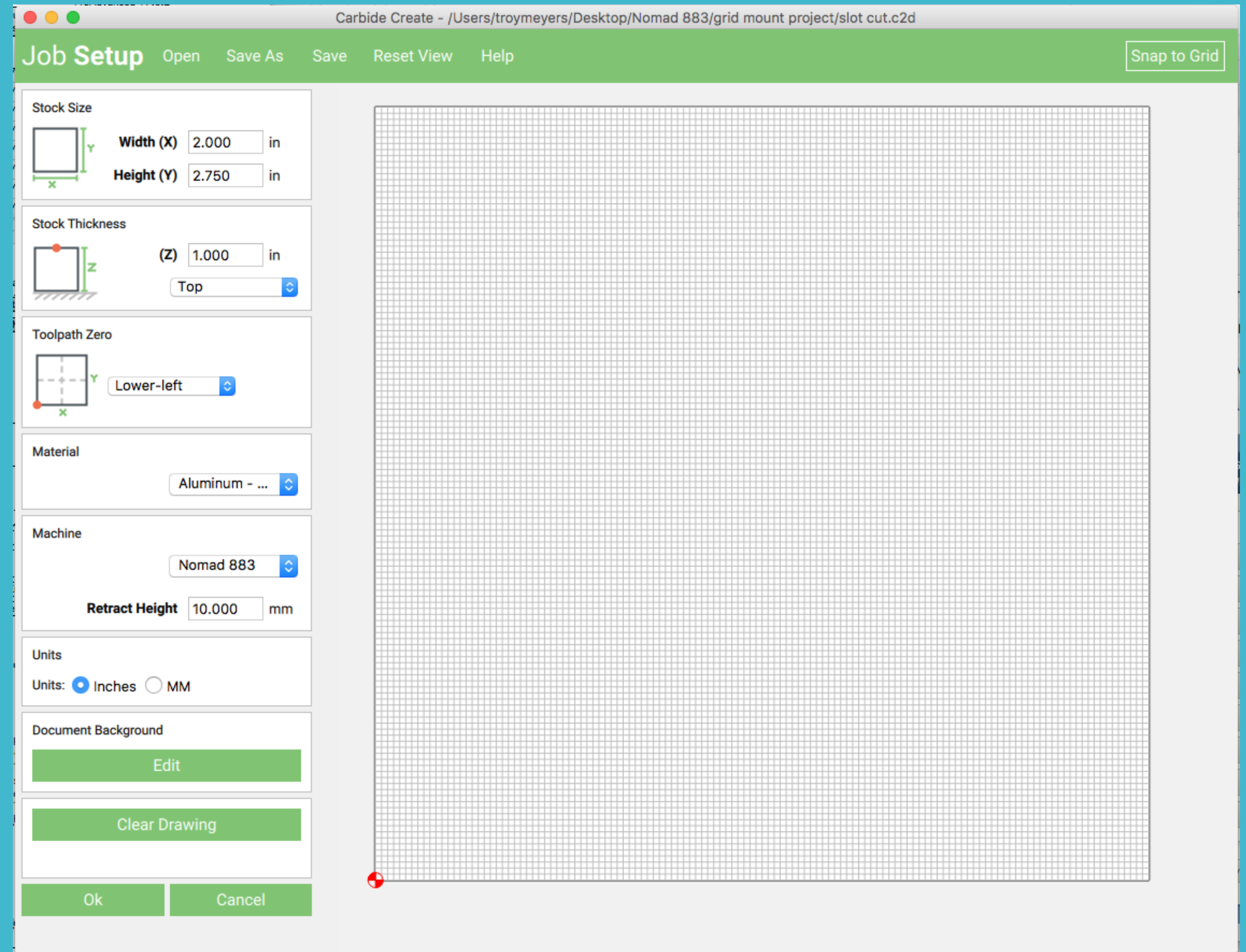
Power input

100 - 240 VAC, 50 - 60 Hz



Software

Section 5



Software Creation

Before we can cut anything on the CNC router we need to create a digital model of what we want to make. The machine can cut 3D models up to a certain depth and also cut or engrave 2D images into materials. Here are some examples of software that you can use.

Carbide Create

This is the native software to our CNC Router. It is fairly limited in what it can do but has a simple interface. <https://carbide3d.com/downloads/>

Fusion360

Complex 3D models can be created relatively easily. Widely used so a lot of training videos are available on YouTube. Allows you to model and then create the cut paths and R code all in one place. <https://www.autodesk.co.uk/products/fusion-360/free-trial> Tool library for fusion : <https://github.com/michaelgrilo/carbide3d-tool-library>

TinkerCad

Simple interface, allows 3D modelling for 3D printers but could be used for CNC. <https://www.tinkercad.com/>

Blender

Steep learning curve for advanced 3D modelling. <https://www.blender.org/>

OpenSCAD

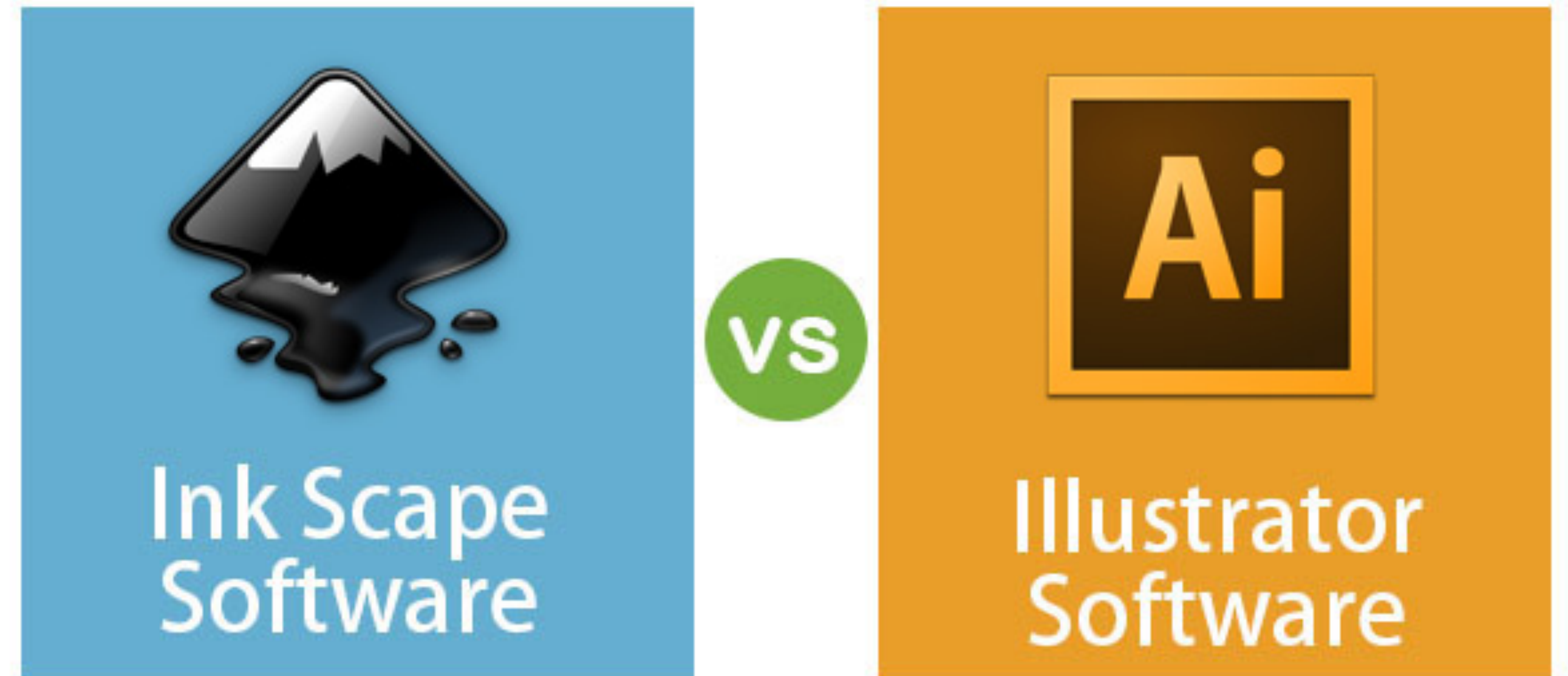
3d software, more complex code based. <https://openscad.org/>

Solid Works

3d software, paid. <https://www.solidworks.com/>

Software Creation

Other software can be used to draw an image and then import it and set the cut paths for use on the CNC Router such as Adobe Illustrator or Inkscape.



Software

Tool Paths

Once a model has been created it needs to be processed ready for the CNC Router by defining the tool paths. This is done within some of the applications on the previous two slide and not in others so the model may need to be imported into Carbide Motion or Fusion 360 to do this.

Each line or section of the model is selected and then the tool that will be used to machine that part is defined along with the type of cut to make. More information can be found here: <https://my.carbide3d.com/carbide-create-course/#toolpaths> to here <https://shapeokoenthusiasts.gitbook.io/shapeoko-cnc-a-to-z/toolpath-basics>

Pocket toolpaths

In a pocket toolpath the end-mill is moved within the boundaries of the shape, to remove all material from the top surface down to a given depth.

Contour/Profile toolpaths

Contour (a.k.a. Profile) toolpaths just follow the contour of a shape, with the option to have the endmill positioned either on the outside or on the inside of the shape, or right on it.

Adaptive clearing toolpaths

Regular pocket and profile toolpaths share a common problem that is not immediately apparent: they involve large tool engagement angles in the material (constantly when slotting, and temporarily in the corners when pocketing), which in turn limits how deep/fast one can cut.

V-Carving toolpaths

V-carving is a specific toolpath strategy dedicated to using V-bits to cut the inside of shapes, resulting in a cut depth that depends on how far apart the sides of the shape are.

Roughing vs. finishing toolpaths

For every toolpath there are two conflicting needs: minimising the total runtime, and getting the best finish quality and dimensional accuracy of the workpiece. Instead of settling for a middle ground that accommodates both constraints, a very common approach is to create two different toolpaths:

The first (roughing) toolpath will be optimised to go fast, maximising material removal rate, but will be programmed to leave a little bit of material around the selected geometry. Even if the tool deflects, vibrates, or more generally produces a poor surface finish, this will be taken care of by the next toolpath. The second (finishing) toolpath will be optimised for getting a good surface finish and accuracy: it can be set to go slower, but even if it isn't, the simple fact that it will have very little material left to cut will reduce the efforts on the tool, and produce a cleaner result.

Software Machining

Once the toolpaths have been defined they can be saved as “G Code”. This is the data that is uploaded to the machine to allow it to cut the model.

The Maker Space uses Carbide Motion software to control the CNC Router. The “G Code” is loaded into Carbide Motion and then the software is used to initialise the machine and set up the start point for cutting and the coordinated of the end-mill. We have step by step instructions to take you through this process.

Creative
DUNBLANE

Instructions

Section 6



Instructions

Setting up for CNC machining a job 1/2

- 1.Position stock piece on CNC bed
- 2.Open Carbide Motion
- 3.Switch on CNC
- 4.Connect to cutter
- 5.Load new file
- 6.Note tool(s) to be used
- 7.Done
- 8.Initialise machine
- 9.Jog / Rapid Position
- 10.Remove extraction unit and fit tool
- 11.Done
- 12.Jog to X/Y coordinates of toolpath zero (note speed increment settings) then set zero for X & Y - Done
- 13.Jog Z to Zero height (top or bottom) & set zero - Done
- 14.Replace extraction unit.

Instructions

Setting up for CNC machining a job 2/2

START MACHINING

- 1.Hit "Run", "Start Job", "Start
- 2.*Tool change required* - Resume
- 3.Switch on router at plug socket
- 4.Set speed as indicated
- 5.Switch router on and close doors
- 6.Start dust extraction unit
- 7.Resume
- 8.Job complete - open cabinet and switch router off then switch off at socket
- 9.Switch off dust extraction unit
- 10.Remove job
- 11.Leave machine clean for next user

Instructions

Setting up for CNC machining a job with BitSetter and BitZero 1/2

1. At the design stage in Carbide Create, on the setup page, set Toolpath Zero to "lower left"
2. Open Carbide Motion, connect to the machine & Initialise
3. Insert Tool..Use the BitZero dowel that matches your first cutting tool..1/4" or 1/8" this will avoid having to change the collet on your first operation. *(If you're using an end mill for your first cut, and 100% accuracy isn't critical, then the end mill can be used as the probe.)*
4. Tool travel to BitSetter
5. Place the BitZero at the bottom left corner of the work piece
6. Jog to position the dowel into the BitZero hole just below the surface and attach the magnetic lead to the collet.
7. Click "PROBE" and select Corner position
8. Begin probe and hold the BitZero in position during its operation
9. The probing operation will now go through a procedure to Zero X, Y & Z
- 10.Return the BitZero to its holder and remove the dowel from the router.
- 11.Run Job, Start and follow setting instructions
- 12.If a tool change is required during the machining process the CNC will travel to the BitSetter and reset Z zero before continuing with the next cut.

Instructions

Setting up for CNC machining a job with BitSetter and BitZero 2/2

Note: There are some cutting tools that cannot be used with the BitSetter, a typical example is the 40mm diameter surfacing tool. In this case, the BitSetter should be disabled before commencing the job. XY zero can still be set with the BitZero but Z zero needs to be manually set.

TO TEMPORARILY DISABLE BITSETTER

1. Power on, connect, but don't initialise yet.
2. Uncheck BitSetter checkbox in the Settings menu.
3. Initialise (the machine will skip the BitSetter probing).

when you are done, go back into the Settings and re-enable the BitSetter.

Instructions

Setting up for CNC machining for a job with a tool change 1/2

When working on a job that requires more than one tool it is necessary to reset the tool Z zero position after a tool change. This would normally be done using Shapeoko's "BitSetter" device. We currently don't have a BitSetter and, until we acquire one, the following procedure should be adopted.

Note: this procedure assumes a basic knowledge of how to operate the CNC with Carbide Motion

1. Complete the design (here I'm assuming Carbide Create is being used) and set the toolpaths
2. Arrange the order of the toolpaths by grouping the different tools together and positioning any paths that will cut right through the stock as final toolpaths. *The order of the toolpaths can be changed by left clicking and moving them up or down the list.*
3. Save the toolpaths using "*in this file*" option and save the file as *filename.c2d*
4. Identify the first toolpath(s) that use the same tool and disable the other toolpaths by right clicking and selecting Disable.
5. Save the file as "*filename tool1*"
6. Close this file and open original *filename* file.
7. Repeat steps 4 & 5 for the next set of toolpaths and save the file as *filename tool2*. Continue for subsequent toolpaths saving as tool3, tool4 etc
8. Position the stock piece on the CMC bed.
9. Open Carbide Motion, switch on the CNC and select "Connect to cutter" then "initialise machine"
10. Select "Jog" from the top menu then select "rapid position" and select front centre **S**
11. Fit the correct tool for this file.
12. Click "Done"

Instructions

Setting up for CNC machining for a job with a tool change 2/2

13. Jog cutter to desired XY position

14. Select "set zero" and click on "Zero X" and "Zero Y". click Done

15. Set "Z" position then select "set zero" and click on "Zero Z" and click Done. *XY&Z have now been zeroed*

16. Select Run from top menu and load "...tool1" file and click Done

17. Click Start Job and click Start

18. Follow on screen instruction

"Tool change required" - just click resume since we have already fitted the first tool.

Switch on router and set speed to that shown

Click Resume

CNC will start cutting and when finished, follow on-screen instructions

19. Click "Load new file" & select *filename tool2*. Click Done

20. Select Jog from top menu then select "Rapid Position"

21. ***This is the most important step...***Select "Rapid to current XY" and the cutter will return to the original XY zero position. Click Done

22. **Switch off router** and Fit tool 2 and repeat step 15 to set new Z position.

23. Click Run & Start Job then click Start.

24. Repeat step 18 and CNC will complete tool2 cuts

25. Follow on screen instructions then repeat from Step 19 for subsequent tool files.